

Introduction: RS³ Tutorial 8 Underground Tunnels

Welcome to RS³. This tutorial demonstrates how to import an Examine3D file and refine the mesh. The model is of underground tunnels under gravity stresses.

The finished product of this tutorial can be found in the **Tutorial 08 Underground Tunnels.rs3dmodel** file. All tutorial files installed with RS3 can be accessed by selecting File \rightarrow Recent \rightarrow Tutorials folder from the RS3 main menu.

CONTENTS

Introduction: RS ³ Tutorial 8 Underground	
Tunnels	1
Step 1: Starting the Model	2
Creating a Blank Document	2
Step 2: Editing the Project Settings	2
Configuring the Units	2
Other Options	2
Step 3: Importing Examine3D	3
Importing The Tunnel Geometry	3
Step 4: Defining the External Box	3
Creating the External Box	3
Step 5: Defining the Materials	3
Defining Materials Properties	3
Step 6: Finalizing the Geometry	4



Dividing All Geometry	4
Step 7: Adding Stress Loading	5
Applying Field Stress To The Model	5
Step 8: Setting Boundary Conditions	5
Adding Model Restraints	5
Step 9: Meshing	5
Configuring and Calculating Mesh	5
Refining the Mesh	6
Step 10: Computing Results	7
Compute	7
Step 11: Interpreting Results	8
Displaying the Results	8
Total Displacement	9

Tutorial Key Concepts Embankment consolidation

Modified Cam Clay Transient groundwater



Step 1: Starting the Model

CREATING A BLANK DOCUMENT



Start RS³ by selecting Programs \rightarrow Rocscience \rightarrow RS3 2.0 \rightarrow RS3 from the Windows start menu. RS³ opens to a blank screen, which allows you to create a new model by pressing the [New Project] button. If the RS³ application window is not already maximized, maximize it now so the full screen space is available for use.



First, save as **Tutorial 08 Underground Tunnels.rs3dmodel**: File \rightarrow Save.

Step 2: Editing the Project Settings

CONFIGURING THE UNITS



Settings

The Project Settings dialog is used to configure the main analysis parameters for your RS³ model. Open the dialog through File \rightarrow Project Settings. This will open the dialog on the first tab: [Units], and set Units = <u>Metric, stress as MPa</u>.

Project Settings: Units Units Units Units Project Set Units Stage Solve	🗊 Project Settings		1	?	×
	Units	Units:	Metric, stress as MPa		-
		m,	m, MN, MN/m, MPa, MN/m3		
	[*] Ω [™] Stress Analysis	Time Units:	Days		-
	- Solver Options	Permeability Units:	Meters/second		•
	- Groundwater	Coordinate:			Ŧ

OTHER OPTIONS

Next, select the [Groundwater] tab.

Project Settings: Ground water

🗊 Project Settings			?	×
Units Units Units Units Units Stages Stress Analysis Units Groundwater	Method: Pore Fluid Unit Weight (MN/m3):	None	•	

Enter Method = <u>None</u> and select [OK] to close the dialog. Go to File \rightarrow Project Summary and enter <u>Underground Tunnels</u> as the Project Title.

Do not change any other settings. Select [OK] to close the dialog.



Step 3: Importing Examine3D

IMPORTING THE TUNNEL GEOMETRY

Select: File \rightarrow Import \rightarrow Import Examine3D file.

Select **UndergroundTunnels.ex3** (from: C:\Users\Public\Documents\Rocscience\RS3 2.0 Examples\Tutorials\Tutorial 08 Underground Tunnels), ensure that when selected in the visibility pane, that in the properties pane for the underground tunnels, the Role = <u>Excavation</u>, and Applied Property = <u>No Material</u>.

Step 4: Defining the External Box

Geology

CREATING THE EXTERNAL BOX

Ensure the <u>Geology</u> tab is selected from the workflow at the top of the screen.

Select: Geometry \rightarrow Create External Box.

A Create External dialog will open, Auto-Expand by = $\underline{0.5}$, press [Auto-Expand by] to update the selection, then [OK].



Step 5: Defining the Materials

Geology

DEFINING MATERIALS PROPERTIES



Under the same tab (Geology) you can assign the materials and properties of our model through Materials \rightarrow Define Materials.

Define Materials The import should have brought the material properties from the Examine3D model, select [Examine3D – AECL Underground Research Laboratory Material], and ensure the following properties are entered in the [Strength] and [Stiffness] tabs respectively



	Material Properties ■		? ×
Examine3D Material	Material 1 Material 2	Name Examine3D - AECL Undergroud Research Laboratory Material	-
Properties: Material 3 Strength Material 4 Material 5 Examine3D - AECL Undergroud Research	Initial Element Loading Field Stress & Body Force Unit Weight (MN/m3): Material Behavior: Drained	0.02	
		Strength Stiffness Staging Datum Dependency Hydraulics Failure Criterion: Hoek-Brown Material Type: Elastic Intact Comp. Strength (MPa): 200 mb Parameter (peak): 31 s Parameter (peak): 1 Material Type:	
Evomino2D	Material Properties	News Examine2D AECI Undergroud Research Laboration Material	? ×
Material Properties:	Material 2 Material 3 Material 4	Initial Element Loading Field Stress & Body Force Unit Weight (MN/m3):	0.02
Stiffness	Examine3D - AECL Undergroud Research L	Strength Stiffness Staging Datum Dependency Hydraulics Type: Linear Isotropic Use Unloading Condition 	
		Loading Young's Modulus (MPa): 60000 Poisson's Ratio: 0.25 Young's Modulus (resid) (MPa): 0	

Step 6: Finalizing the Geometry

Geology

DIVIDING ALL GEOMETRY



Now we can cut into the external box with the tunnels: Geometry \rightarrow 3D Boolean \rightarrow Divide All Geometry.

Geometry

Select the external box in the visibility pane, and in the properties pane change the Role = <u>Geology</u>, Applied Property = <u>Examine3D AECL Underground Research Laboratory</u> <u>Material</u>. Your model should now appear as below:







Step 7: Adding Stress Loading

APPLYING FIELD STRESS TO THE MODEL



Next we go to the <u>Loads</u> tab. This tab allows you to edit the loading conditions. Select: Loading \rightarrow Field Stress.

Field Stress

	Field Stress Type:	Constant			-			OK
	Sigma 1:	20000	ИРа	5	Sigma 2:	20000	MPa	Cancel
eld Stress	Sigma 3:	20000 N	ЛРа					
	Orientation							
	Defined by: Tr	end/Plunge	•					
	Sigma 1: Trend	d:	0	0	Plunge:	-90	0	
	Sigma 3: Trend	i:	90	0	Plunge:	0	0	

Enter Field Stress Type = $\underline{\text{Constant}}$, Sigma 1, 2, and 3 = $\underline{20000}$, Defined by = $\underline{\text{Trend \&}}$ <u>Plunge</u>, Sigma 1 (Trend, Plunge) = $\underline{(0, -90)}$, Sigma 3 (Trend, Plunge) = $\underline{(90, 0)}$, and select [OK].

Step 8: Setting Boundary Conditions

200	+	int	_
162	ua	mu	3

ADDING MODEL RESTRAINTS



Restrain

(Under ground)

Move to the <u>Restraints</u> tab to assign restraints to the external boundary of the model.

RS3 has a built in "Auto Restrain" tool for use on underground models. Select: Restraints \rightarrow Auto Restrain (Underground).

This completes the construction of the model (in terms of geometry).

Step 9: Meshing



CONFIGURING AND CALCULATING MESH

Next we move to the <u>Mesh</u> tab: Mesh \rightarrow Mesh Settings.

🎲 Mesh Settings	5		?	×
Element Type:	4-Noded Tetrahedra	3		-
Mesh Gradation:	Graded			•
# Elements: ~N/A	L .	OK	Can	cel

Enter Element Type = <u>4-Noded Tetrahedra</u>, Mesh Gradation = <u>Graded</u>, [OK]. Then mesh the model: Mesh \rightarrow Mesh. The model should appear as below:



Mesh Settings





REFINING THE MESH



Define Mesh

Next, we want to focus on one tunnel, so we will need to refine the mesh: Mesh \rightarrow Define Refinement Regions.

The box that encapsulates the region of interest is: First Corner (x, y, z) = (-20, -40, 15), Second Corner = (-50, 30, -10), [OK].



The green box will now be replaced by a red one, right-click in the viewport and press [Done]. Enter Element Size = $\frac{2}{2}$ in the Mesh Refinement dialog, [OK].



😚 Mesh Refinemen	t	<i>,,</i> ,		?	\times
Name		Distribution	Element Size	Offset	
Mesh Refine: Ne	w 1	Uniform 🔽	2		0
-			ОК	Can	cel



Graded Mesh

Select Mesh \rightarrow Refine Graded Mesh. The area of interest now has a finer mesh.



Step 10: Computing Results

Compute

COMPUTE



Next, move to the <u>Compute</u> tab. From this tab we can compute the results of our model. First, save: File \rightarrow Save.

Use the Save As dialog to save the file. Next, you need to save the compute file: File \rightarrow Save Compute File. You are now ready to compute the results.



Select: Compute \rightarrow Compute.

Compute



	🚚 RS3 2.0 64-bit		- 🗆 X
Compute Engine	File Queue:	Input File: Tut 8.rs3compute Writing File: Elements: 112261 D	OF: 54573
	Processed Files:	100% (load step Iteration	1 of 1)
	v System Statistics: Free Disk = 711 GB Total Memory = 63.9 GB	Solid Tolerance	ax: 500
	Execution Priority: Normal	Fluid Tolerance Ma	ax:
	Information: Read solid restraintsDone Filling initial stress Solid elements Bolt elements Beam and strut elements Done		^
	Time to read file: 1.70207	Pause	Abort
	Computing, Please Wait		00:00:02

Step 11: Interpreting Results

Results

DISPLAYING THE RESULTS



Next we move to the <u>Results</u> tab. From this tab we can analyze the results of our model. First, refresh the results: Interpret \rightarrow Refresh Results.



On the top right corner of the Results tab, you should see two drop down menus:

0	Solids 🔻
°ĭ	Sigma 1 Effective 🔻



We will analyze a number of different "Data Type" results. Turn on the exterior contours such that we can see some results: Interpret \rightarrow Show Excavation Contour.



We also want to define a plane that goes through the slanted cross tunnel. First, we must define a plane, Interpret \rightarrow XY Plane. In the Create Plane dialog, enter: Center (x, y, z) = (-16.0675, -13.7231, 1.941122), Normal (x, y, z) = (0, 0, 1), then press [OK].



x y z Plane Center: -16.0675: -13.7231 1.941122 Plane Orientation Defined by: Normal Vector • X: 0 Y: 0 Z: 1 Normalized: [0, 0, 1]		😚 Create Plane		?	×
OK Cancel	Create Plane	x Center: -16.0675! Plane Orientation Defined by: X: 0 Y: Normalized: [0, 0, 1]	y -13.7231 Normal 1 0 Z:]	z 1.94112 Vector	22

TOTAL DISPLACEMENT

In the top right corner of the Results tab, ensure $Element = \frac{Solids}{N}$, and change Data Type = $\frac{Total Displacement}{N}$:



The Total Displacement results are shown below.



Other results are available to view as well. Thank you, this concludes the tutorial.